Thermal Induced Stresses on a PCB

Problem Description

This tutorial describes the steps to perform a thermal-structural simulation of a printed circuit board using ANSYS SpaceClaim and ANSYS Mechanical.

The tutorial shows you how to 1.) prepare a geometry in ANSYS SpaceClaim so that it works in cooperation with the Trace Mapping feature of ANSYS Mechanical and 2.) demonstrates the use of the Trace Import feature by examining the warpage (deformation) of a simply supported printed circuit board (PCB) as a result of uniform thermal loading.

Features Demonstrated

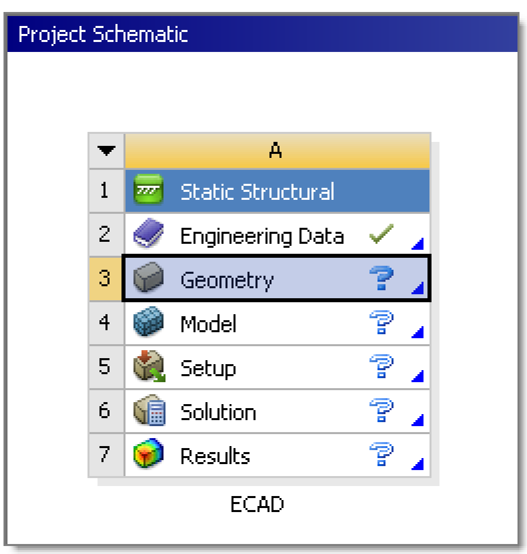
- Engineering Data/Materials
- Static Structural Analysis
- Electronic Computer-Aided Design (ECAD)
- Trace Mapping

Procedure

1. **Set Up the Analysis**.

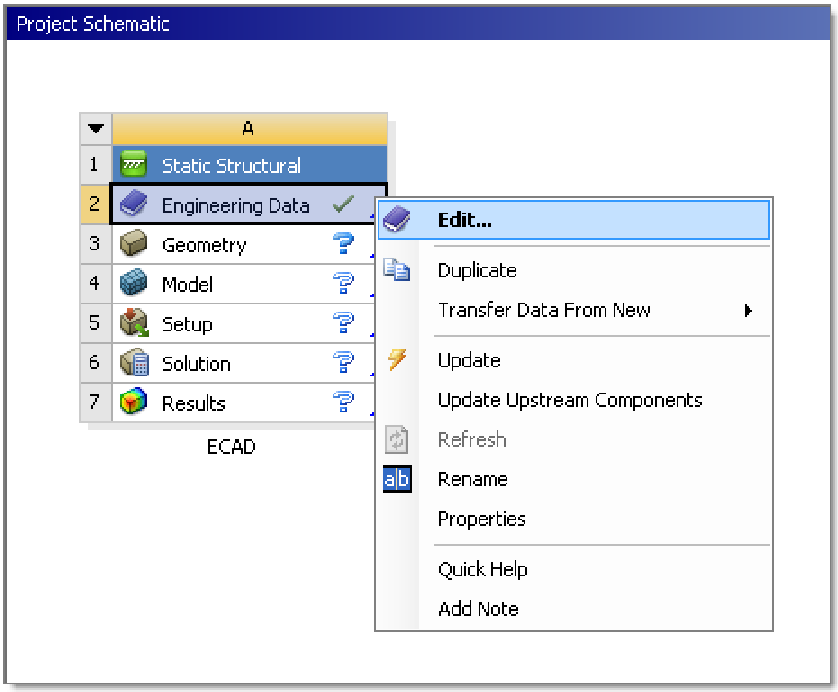
   a. Open ANSYS Workbench.

   b. Create a Static Structural analysis and name it "ECAD." You may create any name as desired.

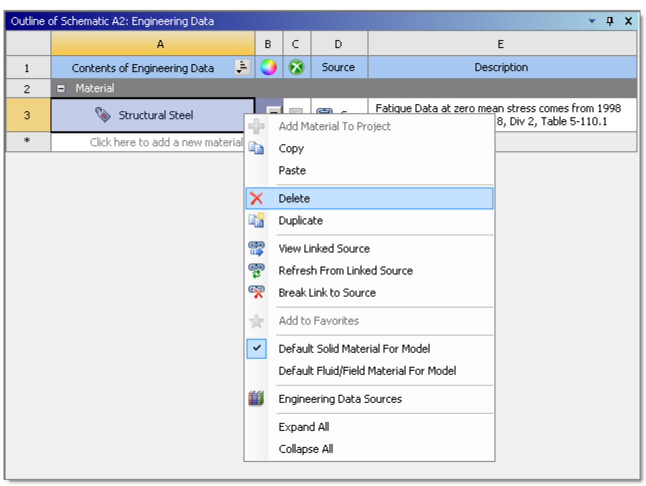


2. **Select Materials in Engineering Data**.

   a. Open the Engineering Data Workspace: right-click on the **Engineering Data** cell and select **Edit**.

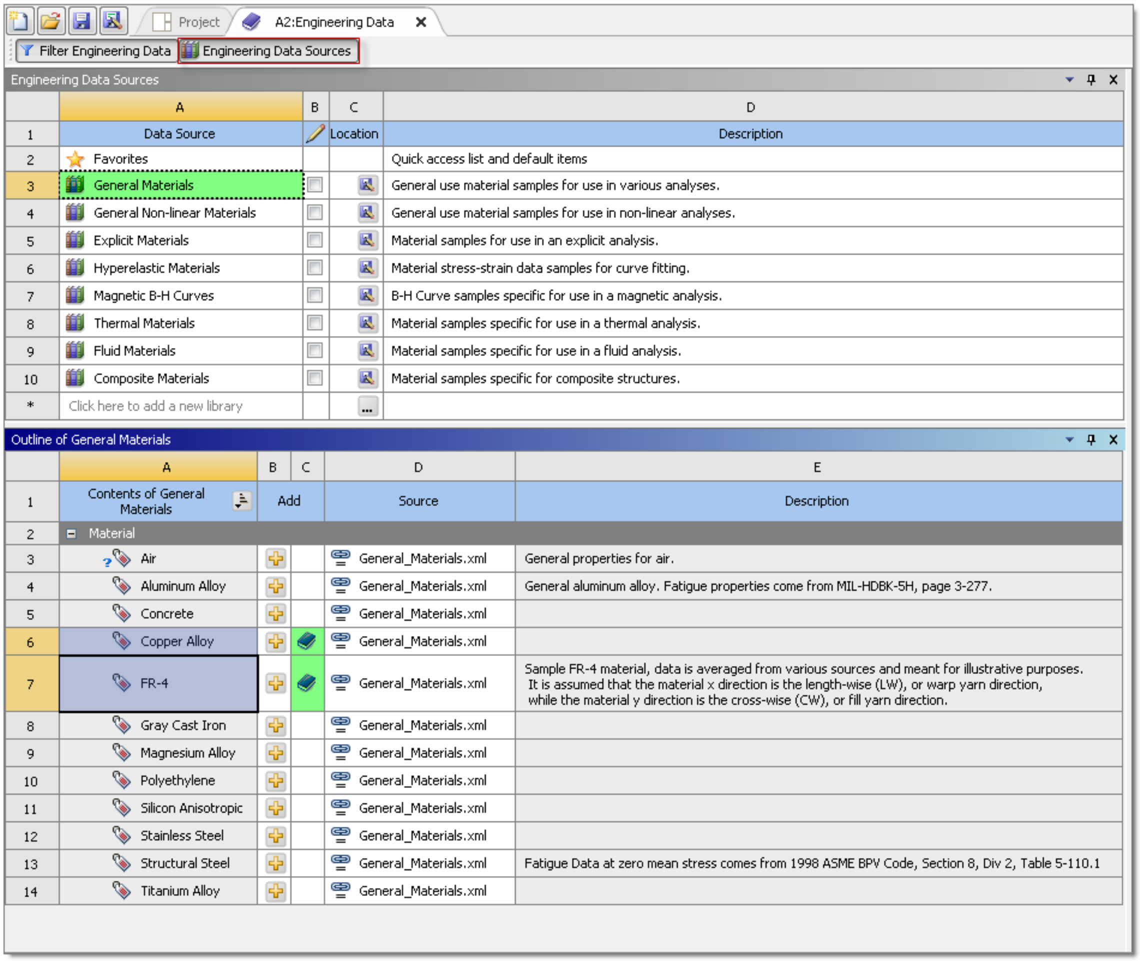


   b. Select and delete **Structural Steel** (right-click>**Delete**), the default material.



   c. Select the **Engineering Data Sources** button on the toolbar.

   d. From the **General Material** library, add **FR4** (Dialectic Material) and **Copper Alloy** using the plus sign button in the **Add** column. A book icon displays in the **Add** column when you select the material.



Project | A2:Engineering Data

Filter Engineering Data | Engineering Data Sources

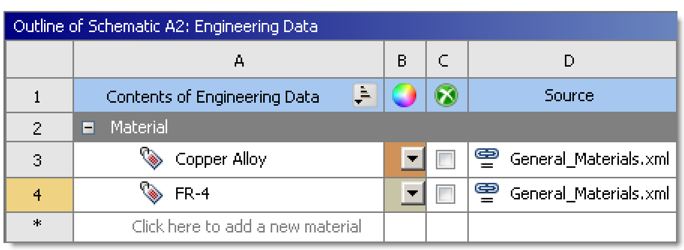
Engineering Data Sources

	A	B	C	D
1	Data Source		Location	Description
2	Favorites			Quick access list and default items
3	General Materials			General use material samples for use in various analyses.
4	General Non-linear Materials			General use material samples for use in non-linear analyses.
5	Explicit Materials			Material samples for use in an explicit analysis.
6	Hyperelastic Materials			Material stress-strain data samples for curve fitting.
7	Magnetic B-H Curves			B-H Curve samples specific for use in a magnetic analysis.
8	Thermal Materials			Material samples specific for use in a thermal analysis.
9	Fluid Materials			Material samples specific for use in a fluid analysis.
10	Composite Materials			Material samples specific for composite structures.
*	Click here to add a new library		...	

Outline of General Materials

	A	B	C	D	E
1	Contents of General Materials	Add		Source	Description
2	Material				
3	Air			General_Materials.xml	General properties for air.
4	Aluminum Alloy			General_Materials.xml	General aluminum alloy. Fatigue properties come from MIL-HDBK-5H, page 3-277.
5	Concrete			General_Materials.xml	
6	Copper Alloy			General_Materials.xml	
7	FR-4			General_Materials.xml	Sample FR-4 material, data is averaged from various sources and meant for illustrative purposes. It is assumed that the material x direction is the length-wise (LW), or warp yarn direction, while the material y direction is the cross-wise (CW), or fill yarn direction.
8	Gray Cast Iron			General_Materials.xml	
9	Magnesium Alloy			General_Materials.xml	
10	Polyethylene			General_Materials.xml	
11	Silicon Anisotropic			General_Materials.xml	
12	Stainless Steel			General_Materials.xml	
13	Structural Steel			General_Materials.xml	Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
14	Titanium Alloy			General_Materials.xml	

e. Select the **Engineering Data Sources** button. The new materials display in the **Outline of Schematic Pane** and will now be available in Mechanical.



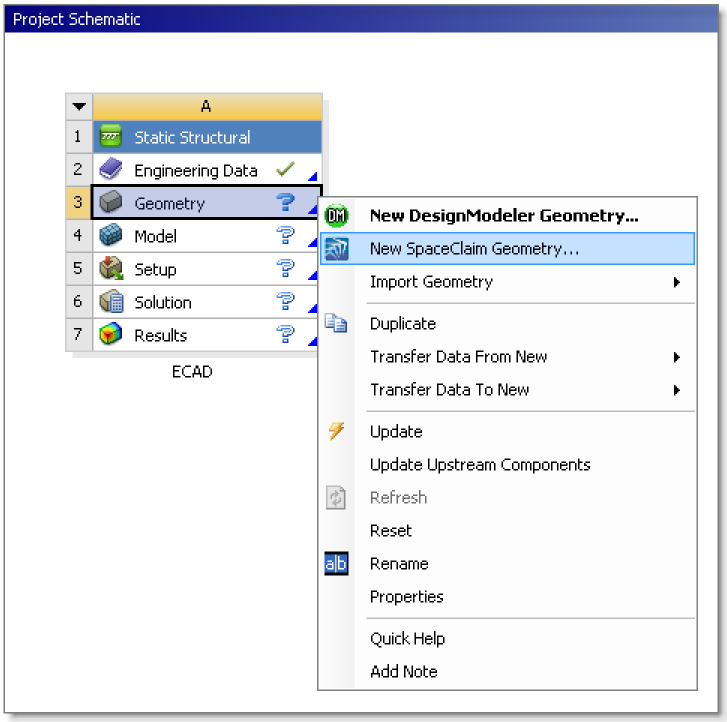
Outline of Schematic A2: Engineering Data

	A	B	C	D
1	Contents of Engineering Data			Source
2	Material			
3	Copper Alloy			General_Materials.xml
4	FR-4			General_Materials.xml
*	Click here to add a new material			

f. Return to the Workbench Project page.

3. **Define Geometry**.

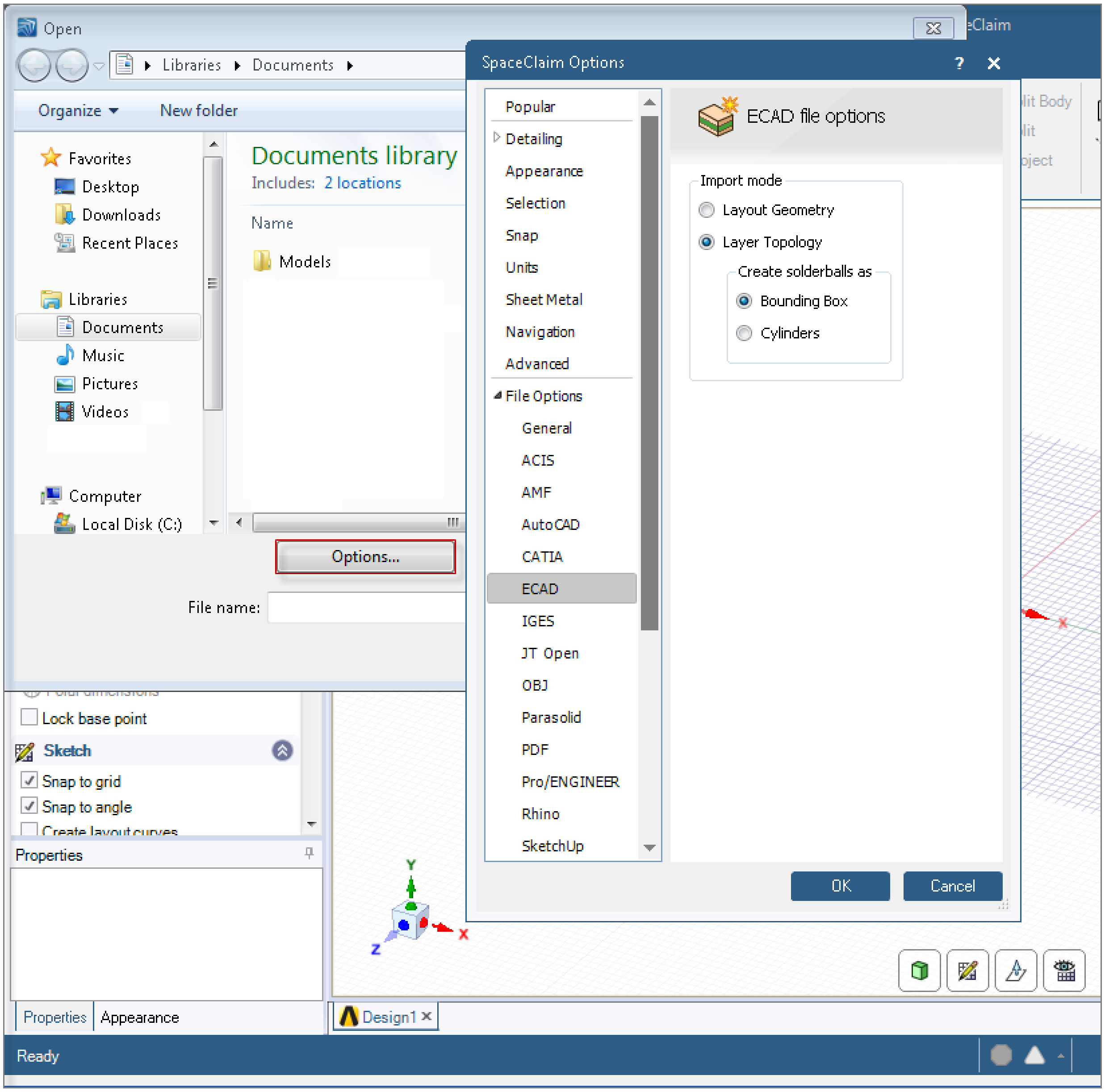
   a. Right click on the **Geometry** cell and select **Edit Geometry in SpaceClaim**.



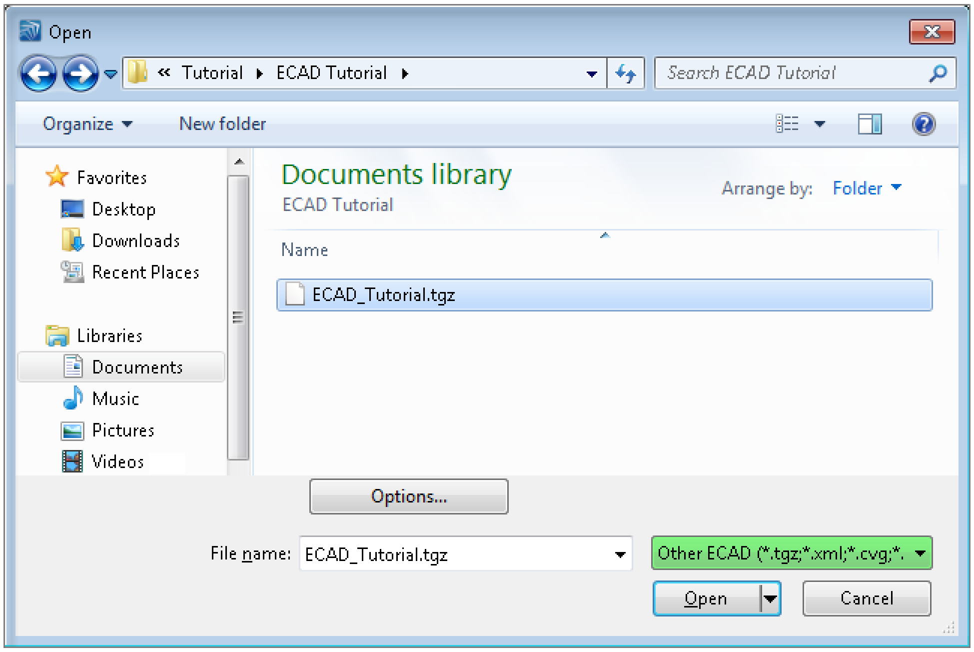
The SpaceClaim application opens.

b. In SpaceClaim, select **File**>**Open**.

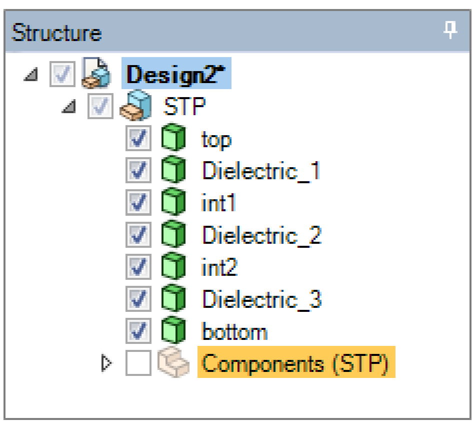
c. From the **Open** dialog box, select the **Options** button and verify that the Layer Topology option is selected under **File Options**>**ECAD**. Click **Ok**.



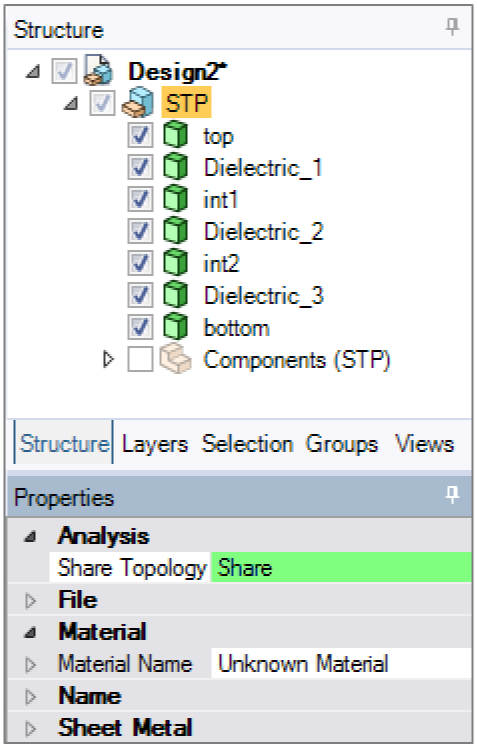
d. From the **Open** dialog box, import the geometry file provided: file name `ECAD_Tutorial.tgz`. This file is available on the ANSYS Customer Portal.



e. Uncheck the **Components (STP)** object in the tree. They will not be needed in Mechanical. Select and view the geometries as desired.



f. Select the **STP** object and then set the **Share Topology** property to **Share**.

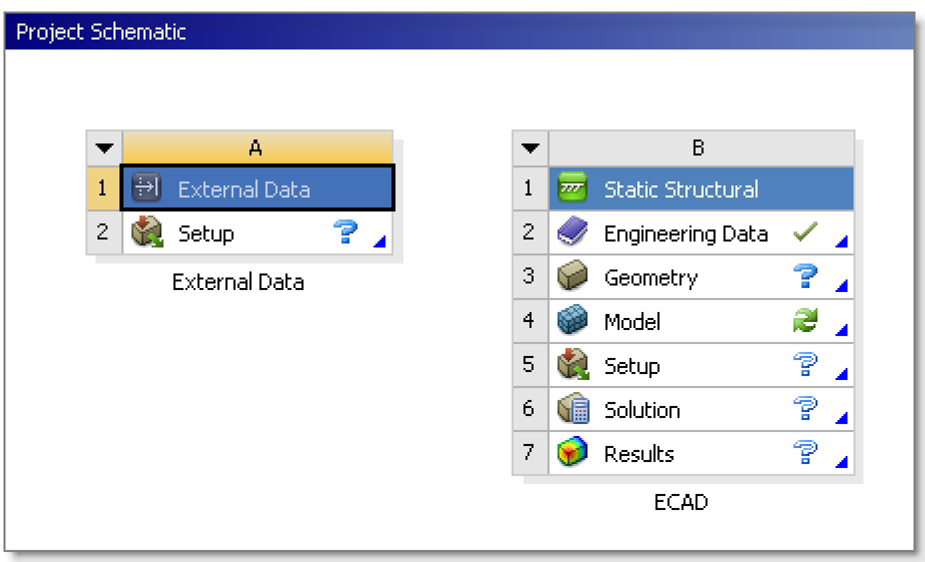


g. Save the file as `ECAD_Tutorial_File.scdoc`.

h. Return to the Workbench Project page.

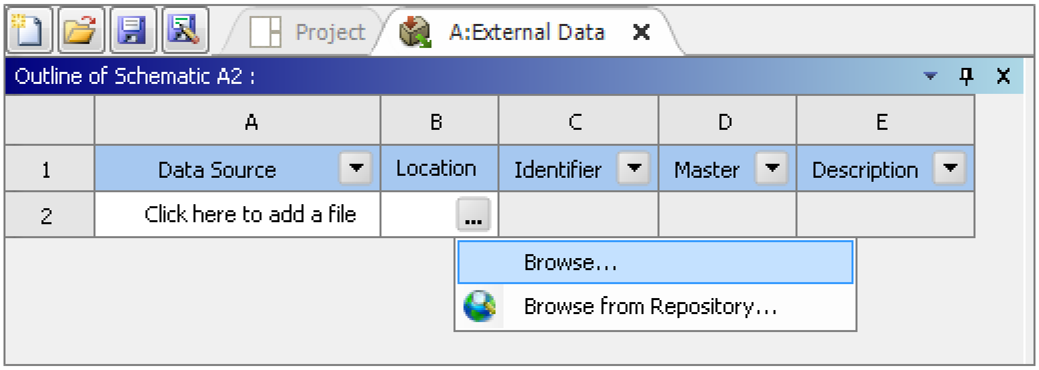
4. **Import the Geometry into Mechanical**.

a. Place an **External Data** system into the project and drag it in front of the Static Structural system.

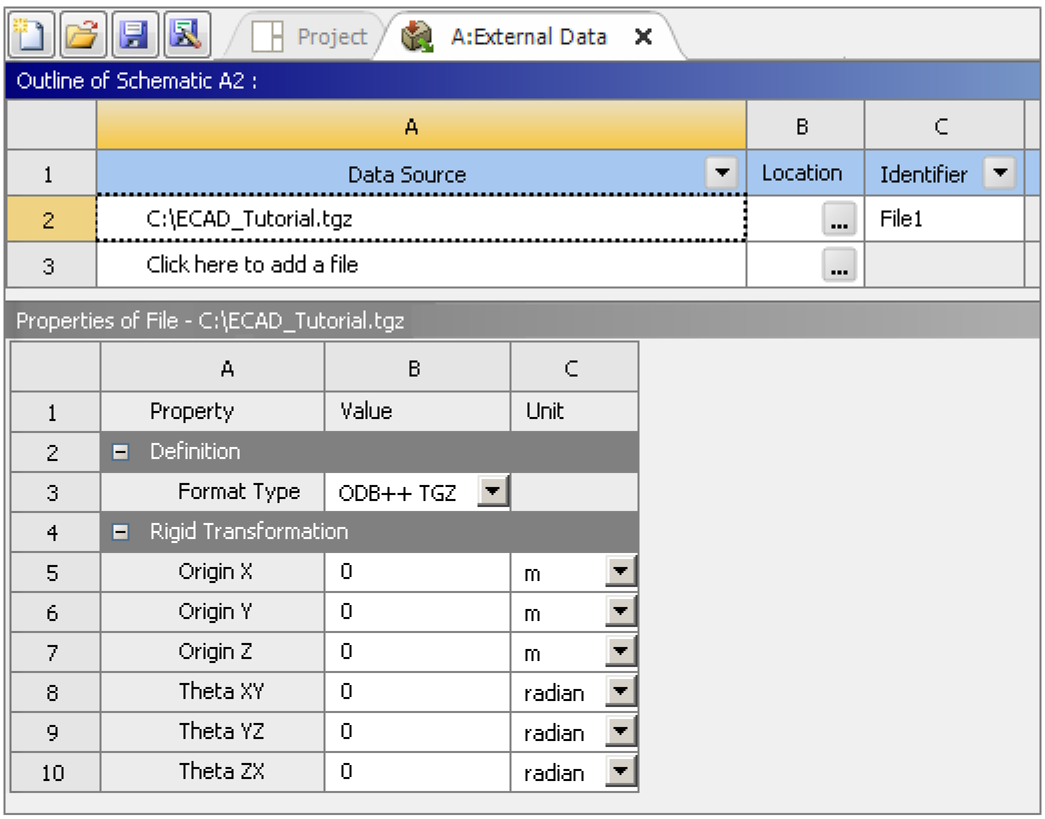


b. Right-click on the **Setup** cell and select **Edit**.

c. Select the button in the **Location** column, browse to the `ECAD_Tutorial.tgz` file, and open it.



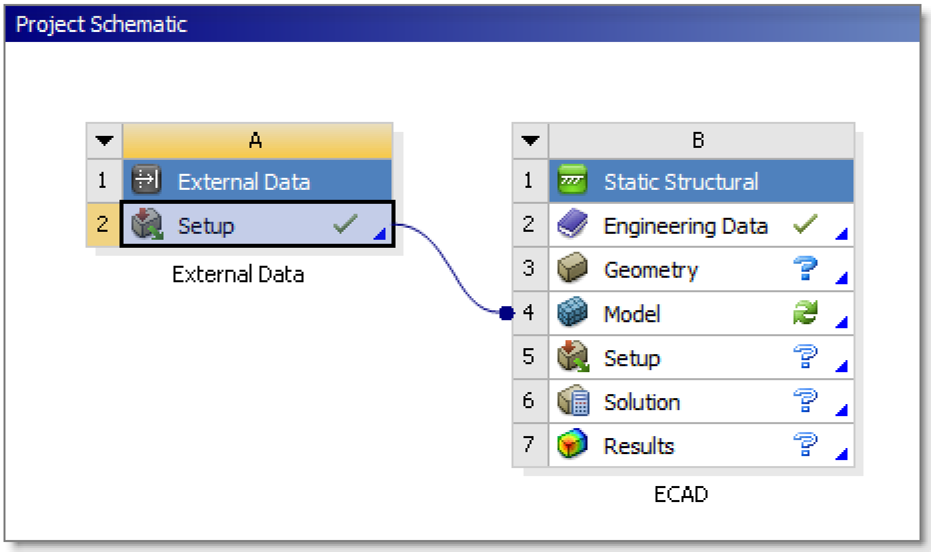
d. Select the row for the file in the **Outline** to display the properties. As needed, specify **ODB++TGZ** for the **Format Type**. Note the default Identifier, **File1**.



e. Return to the Workbench Project page.

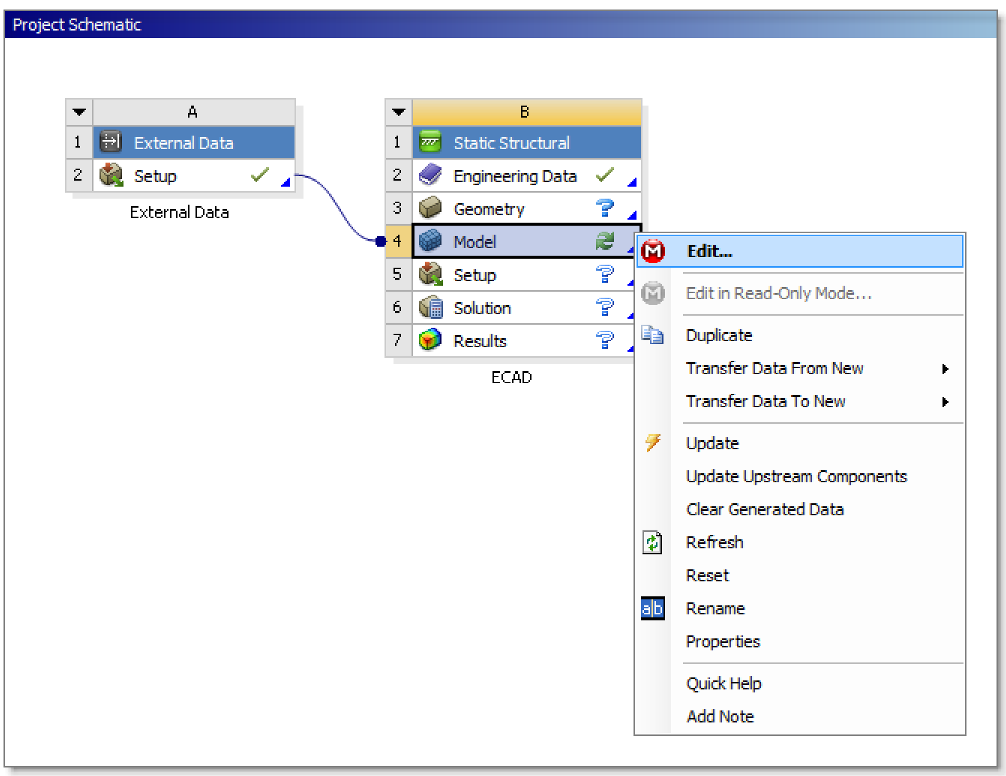
f. Select the **Update Project** button.

g. Drag and drop the **Setup** cell onto the **Model** cell of the Static Structural system to create a link.



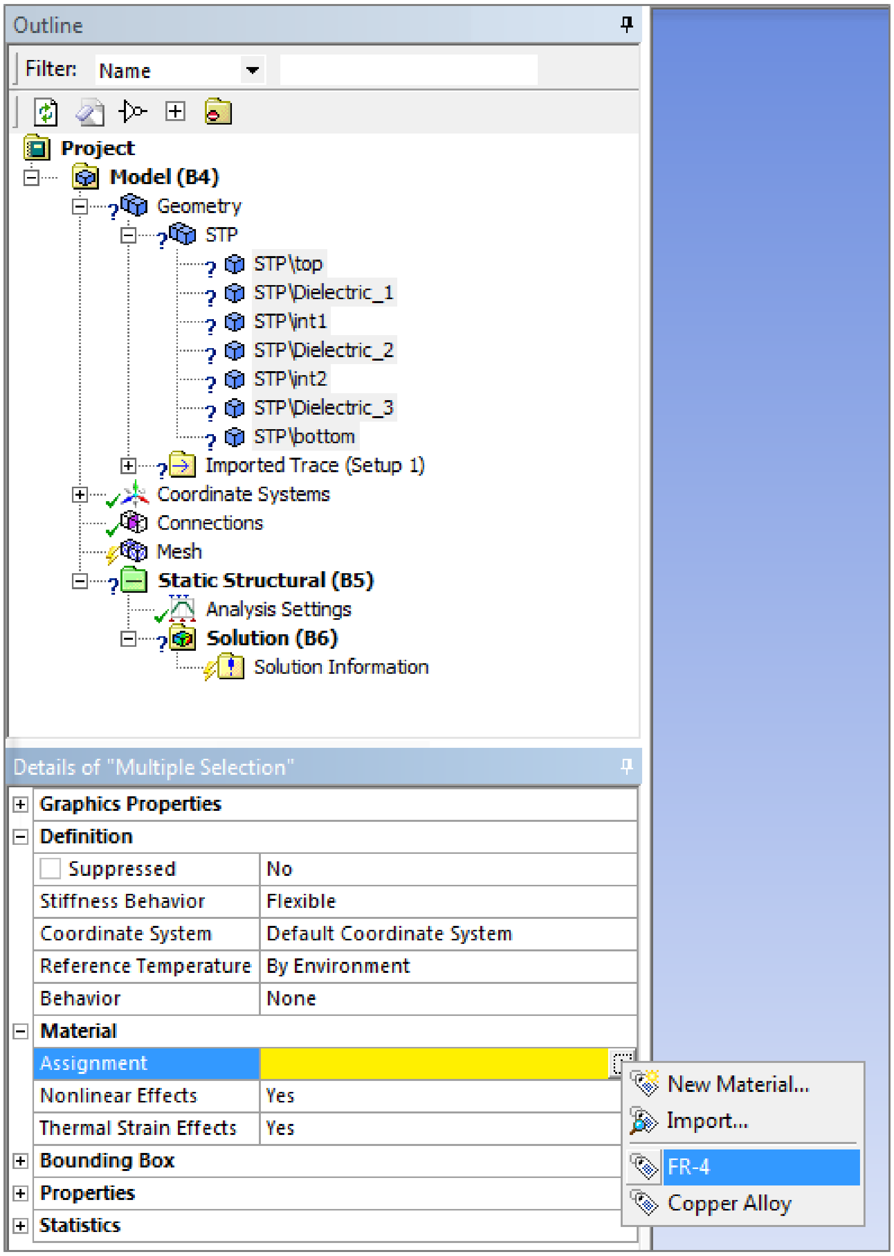
h. Right-click on the **Geometry** cell and browse (**Import Geometry**>**Browse**) to the geometry file you saved in SpaceClaim (`ECAD_Tutorial_File.scdoc`) and open it.

i. Right-click on the **Model** cell and select **Edit** to open the files in Mechanical.



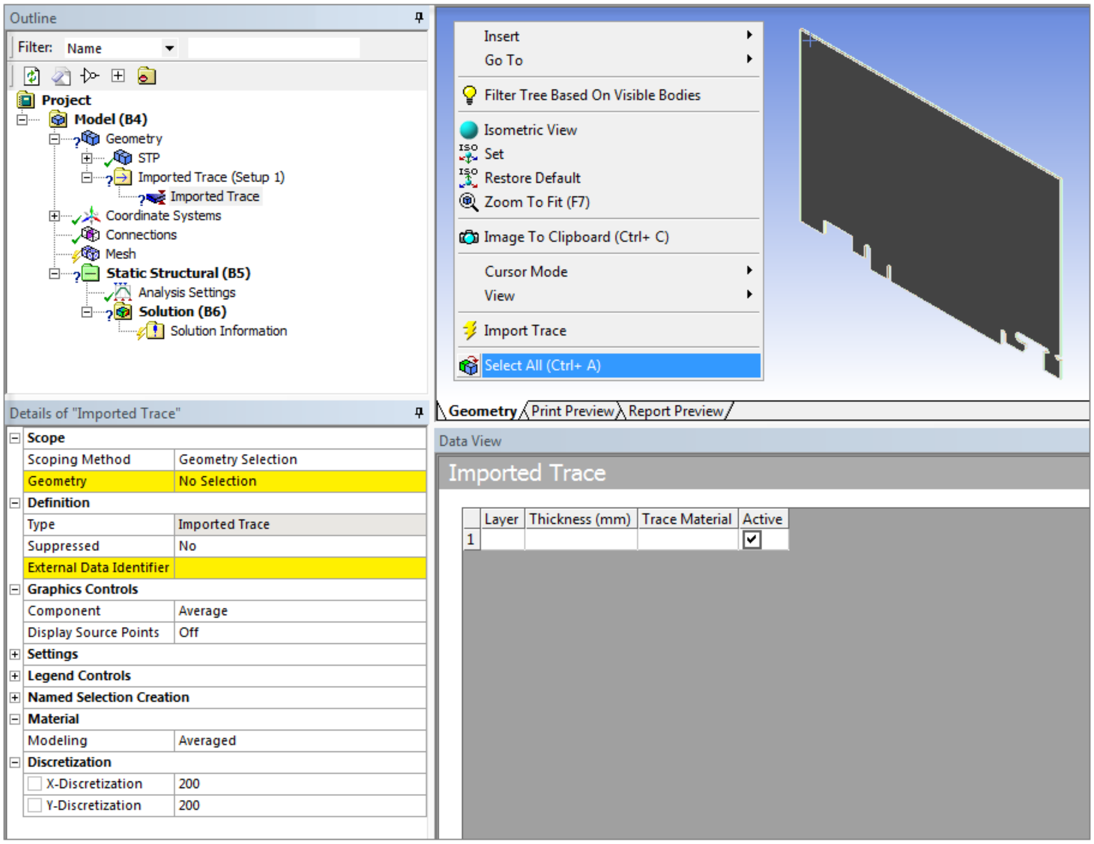
5. **Specifying Materials.** In Mechanical you will note that the Geometry object is underdefined.

   a. Open the **STP** object and select all of the child objects. Select **FR-4** from the drop-down list of the **Assignment** property.

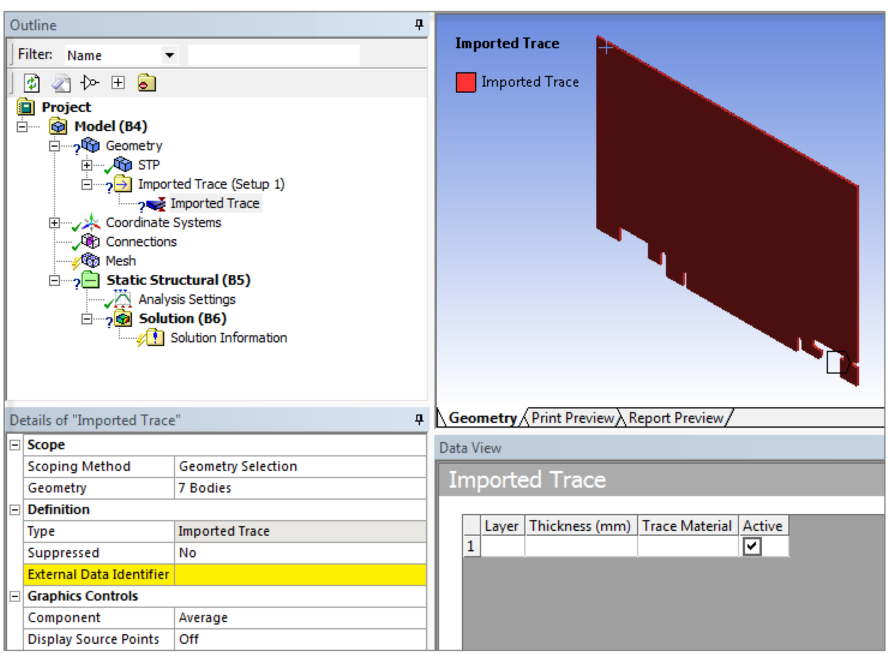


b. Open the **Imported Trace** folder and select the **Imported Trace** object.

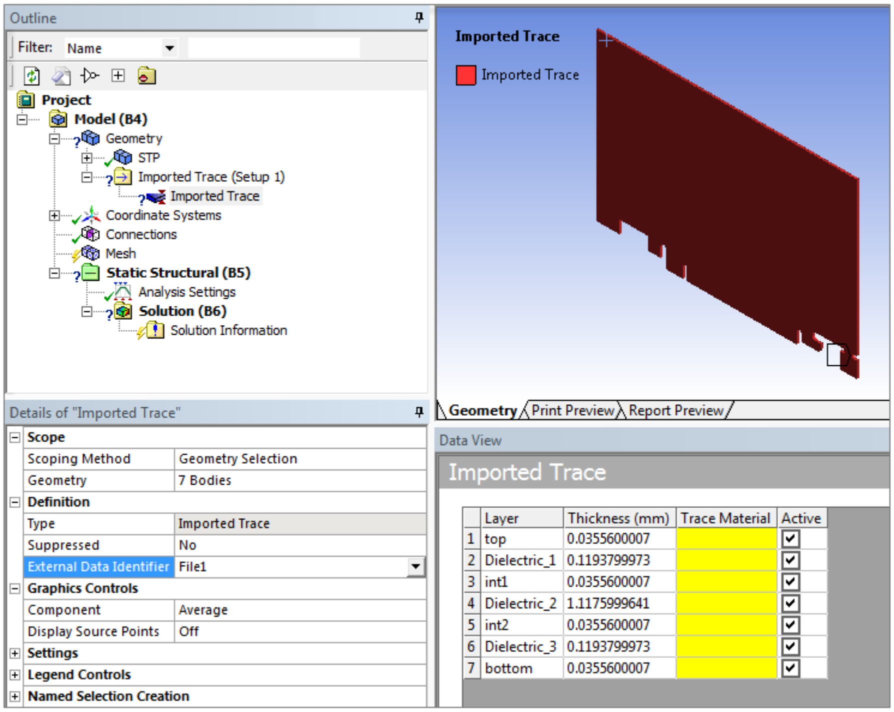
c. Right-click in the Geometry window and select the option **Select All**.



d. Click the **Geometry** property in the Details view and click **Apply**. Seven bodies are specified for the **Geometry** property.



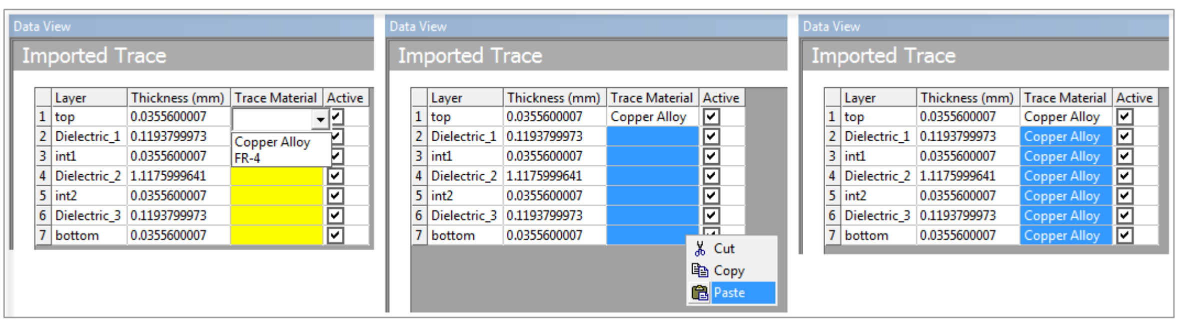
e. Select **File1** (from the External Data system) for the **External Data Identifier** property.



f. In the **Imported Trace Data View**, select the **Trace Material** field and specify **Copper Alloy**.

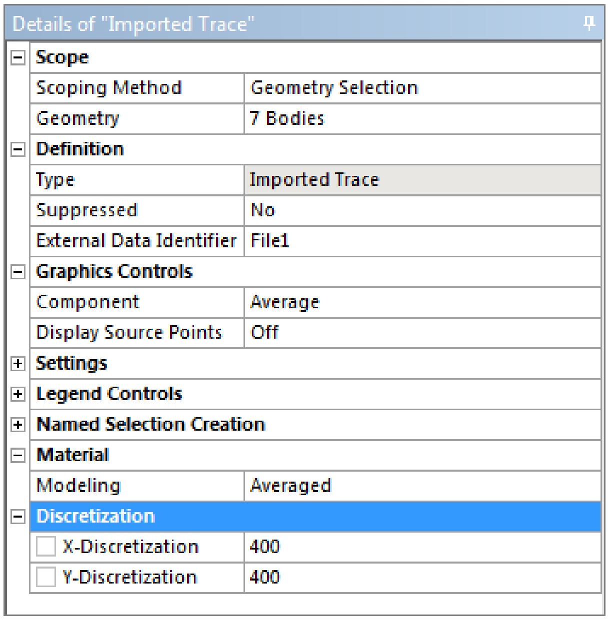
g. Once specified, right-click on the field again and select **Copy**. Select the remaining Trace Material fields using the Shift key, right-click again, and select paste.

All of the remaining cells populate with the **Copper Alloy** material.

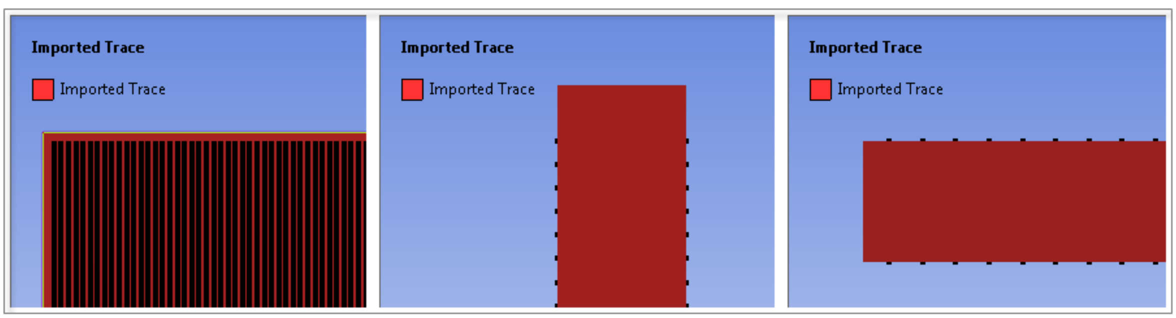


6. **Define Trace Properties**.

a. Specify the **X-Discretization** and **Y-Discretization** properties as **`400`**.



b. Set the **Display Source Points** property to **On** to view the alignment of the source points provided by the trace layout files. Rotate the model and zoom in to view the points. Once you have finished, return the **Display Source Points** property to the **Off** setting.



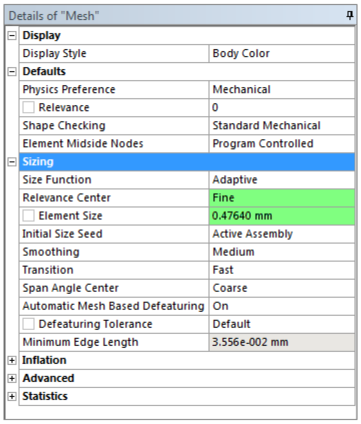
Note

If you ever encounter misaligned source points in a simulation, you can use the Rigid Transformation controls in the External Data system to align the source mesh with the target.

7. **Define Mesh Properties**.

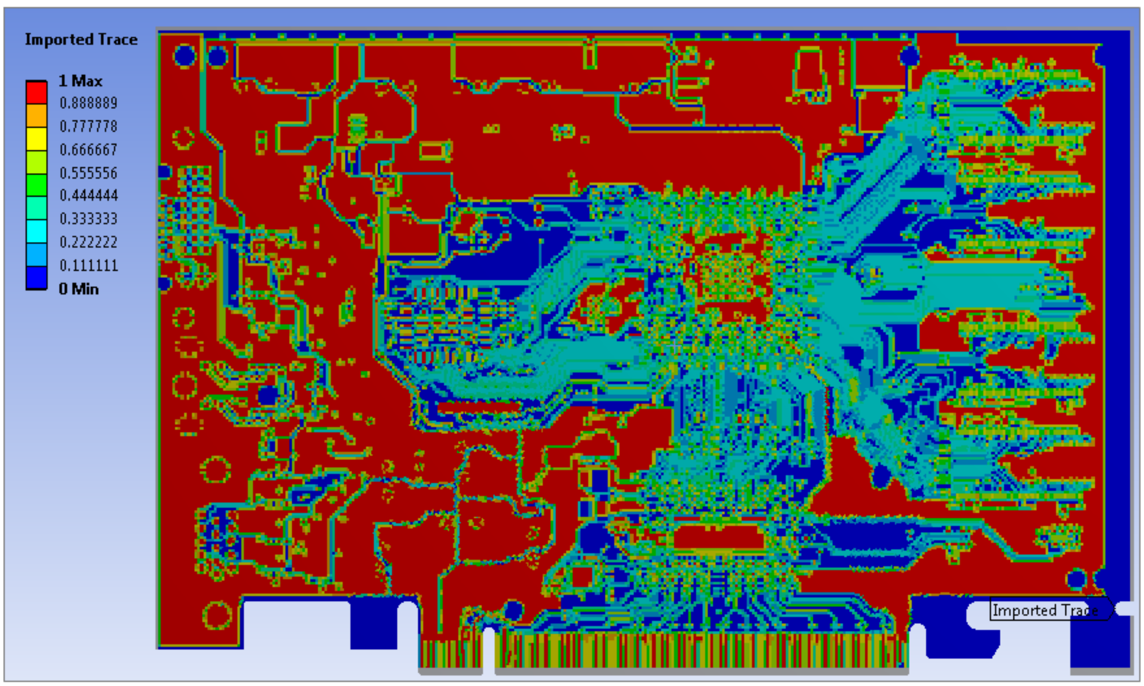
   a. Select the **Mesh** object.

   b. Under the **Sizing** category, specify the **Relevance Center** property as **Fine** and the **Element Size** as `0.47640` (mm). These actions refine the mesh.



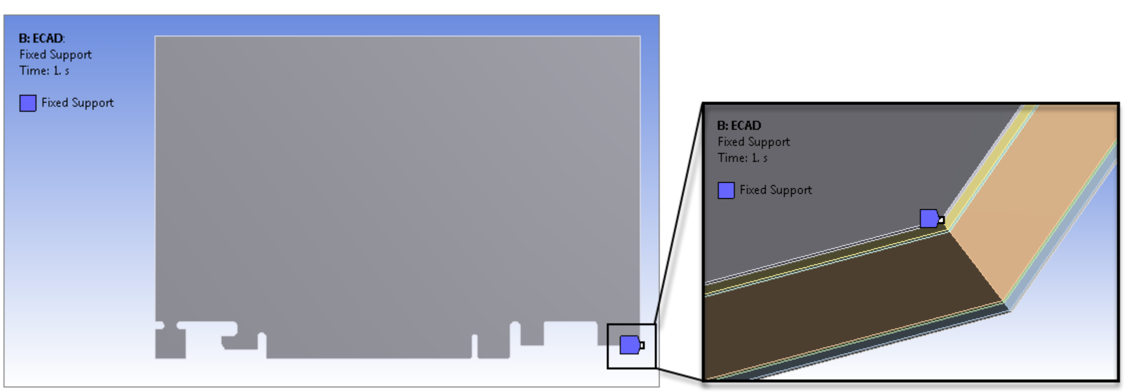
Details of "Mesh"	
Display	
Display Style	Body Color
Defaults	
Physics Preference	Mechanical
Relevance	0
Shape Checking	Standard Mechanical
Element Midside Nodes	Program Controlled
Sizing	
Size Function	Adaptive
Relevance Center	Fine
Element Size	0.47640 mm
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Automatic Mesh Based Defeaturing	On
Defeaturing Tolerance	Default
Minimum Edge Length	3.556e-002 mm
Inflation	
Advanced	
Statistics	

c. Right-click on the **Imported Trace** object and select the **Import Trace** option. This mapping process will take several moments to complete. Once complete the mapping should appear as illustrated in the following image.

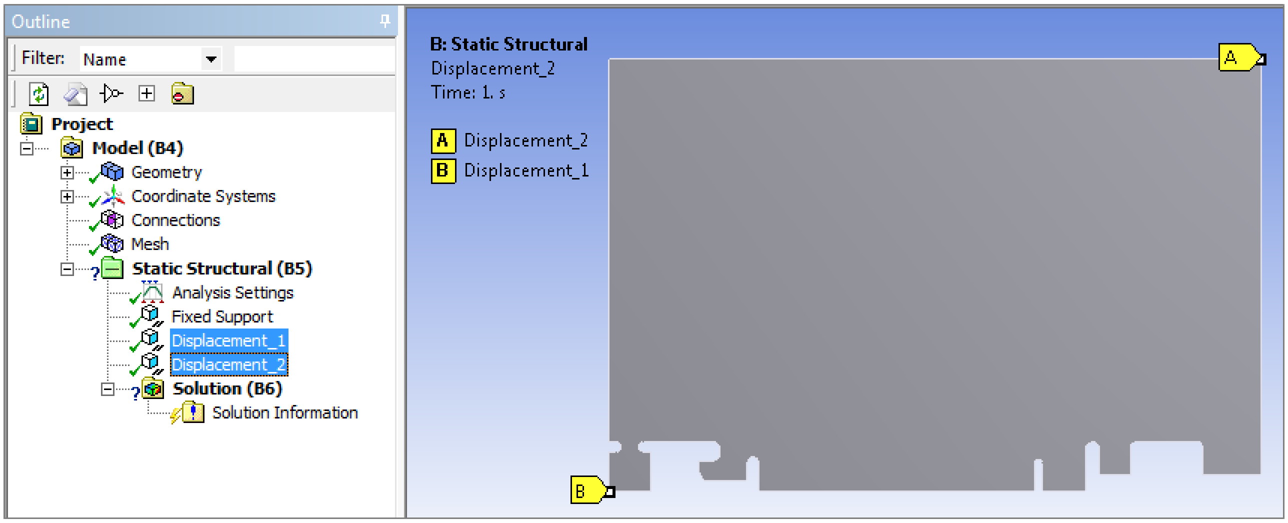


8. **Specify Boundary Conditions**.

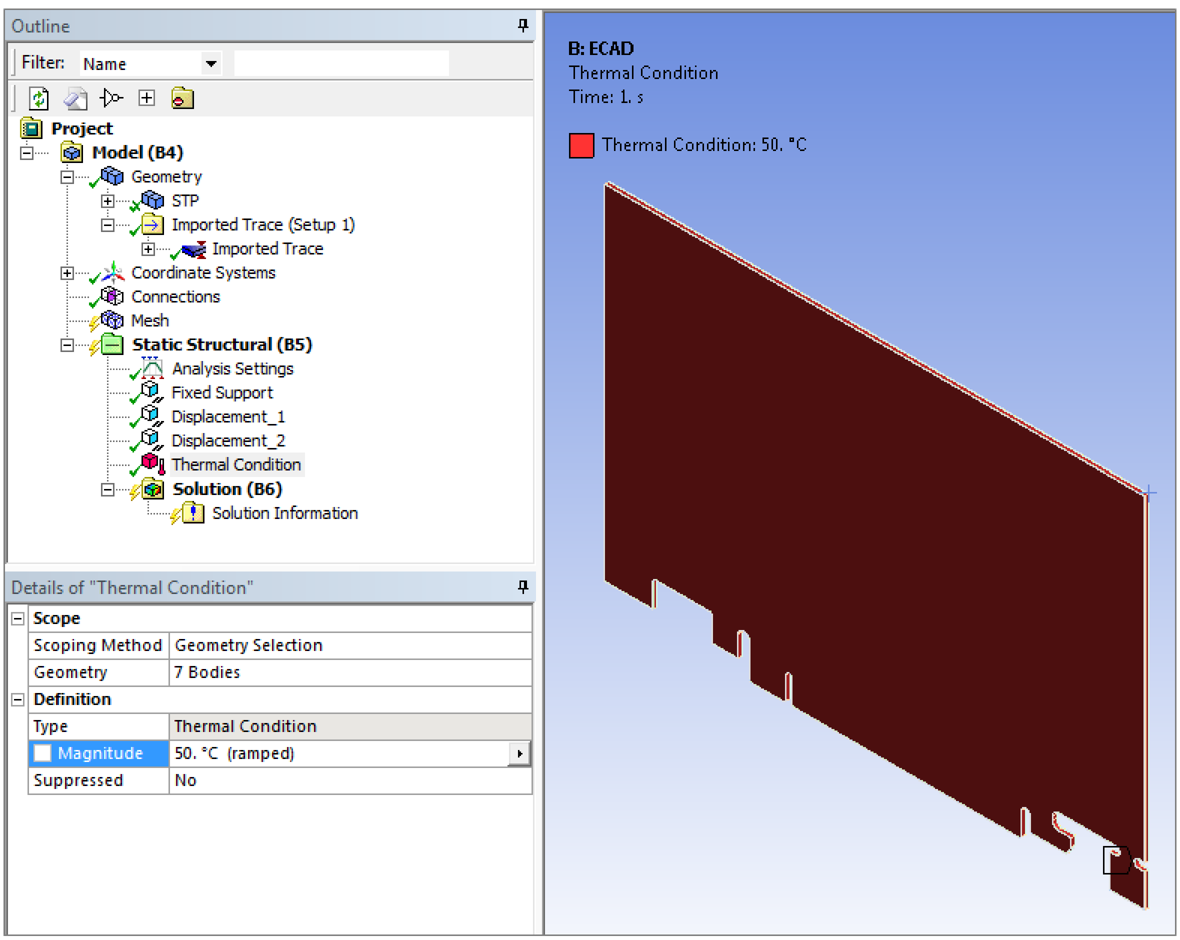
a. Select the vertex from the bottom corner illustrated below and add a **Fixed Support**.



b. Apply **Displacement** boundary conditions to the bottom two corners illustrated below. Specify a `0mm` displacement for the **Y Component** and **Z Component** of the first displacement and a `0mm` displacement for the **Z Component** of the second displacement.



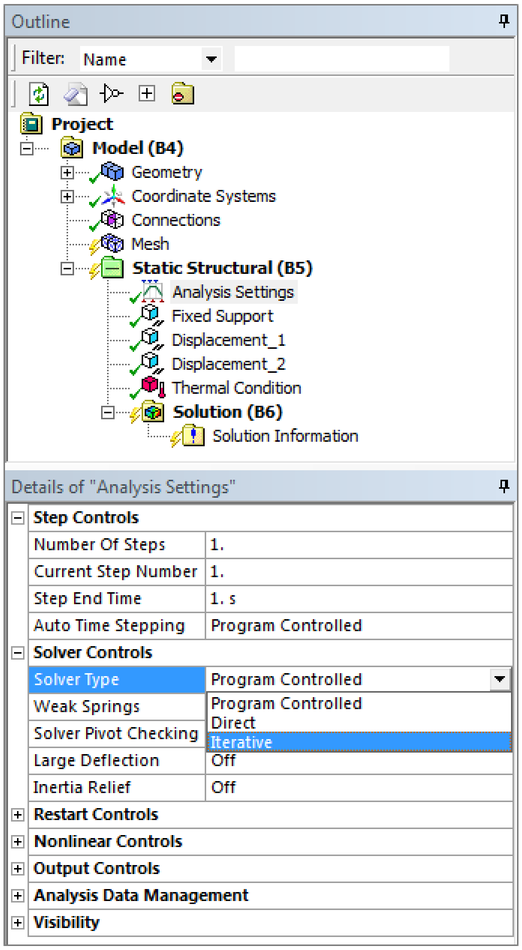
c. Apply a **Thermal Condition** load to all bodies (**Ctrl**+**A**) of the model and specify a temperature magnitude of `50`°.



9. **Specify Solver Type**.

a. Select the **Analysis Settings** object.

b. Set the **Solver Type** property to **Iterative**.

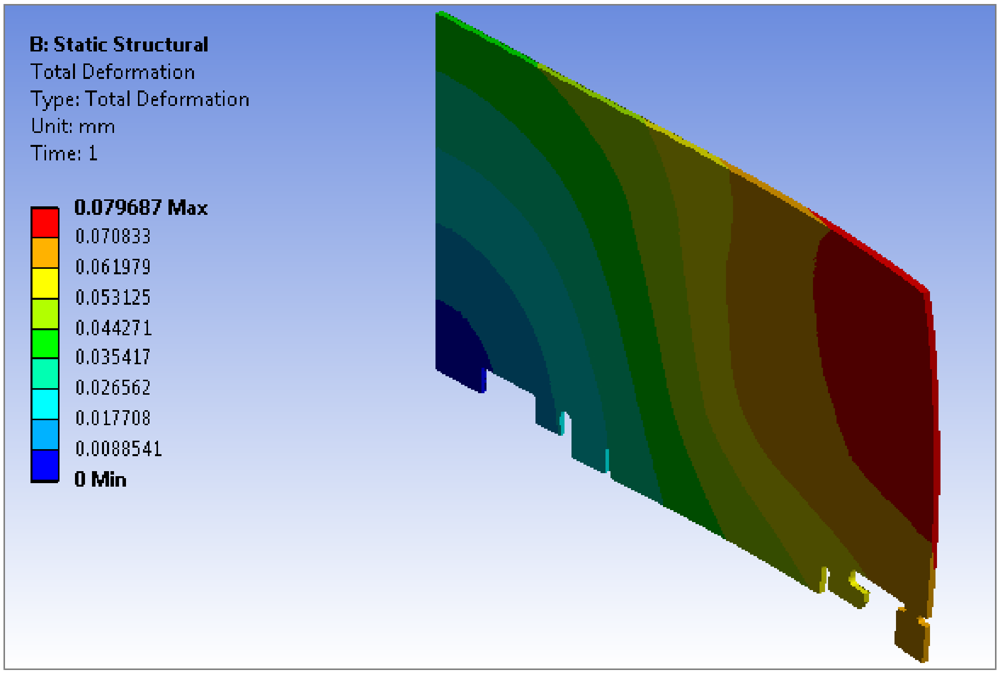


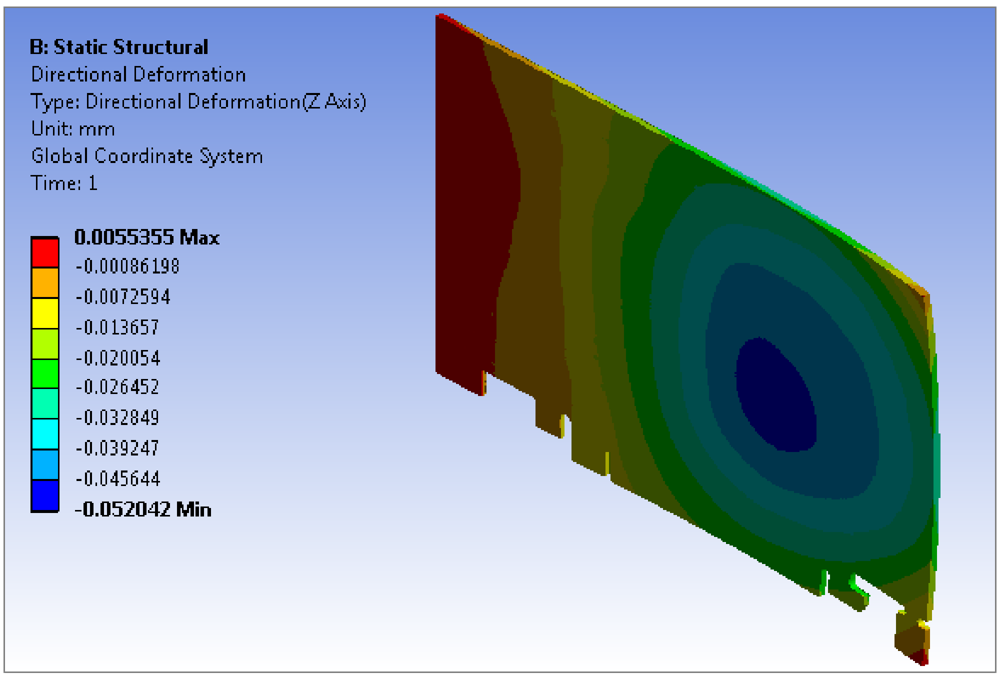
10. **Generate Solution and Define Results**.

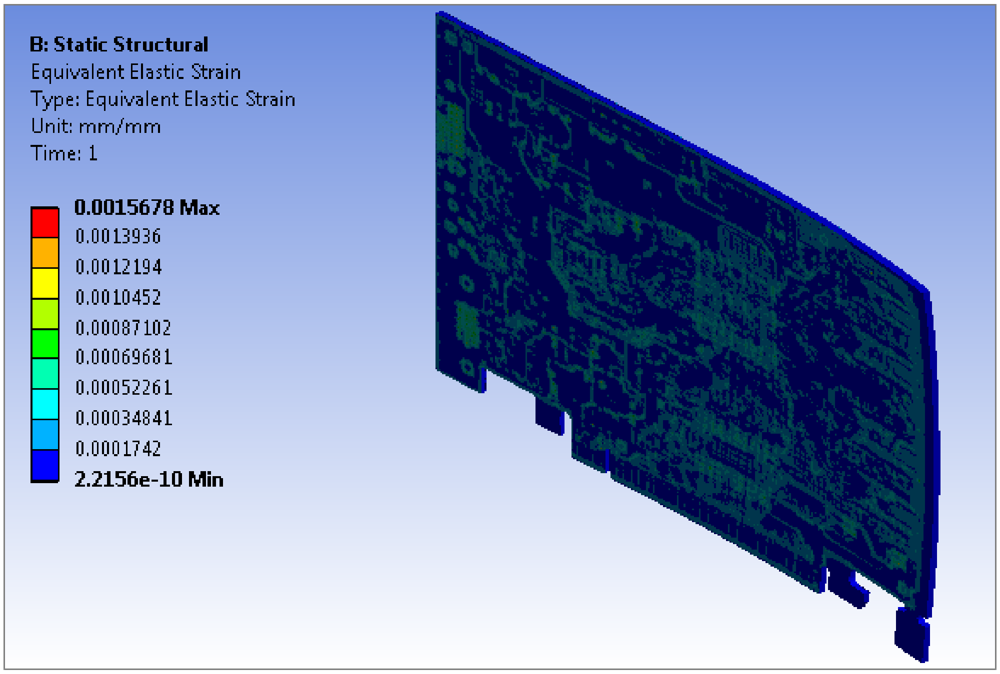
a. Solve the analysis. This process will take several minutes.

b. Apply results as desired.

Total Deformation, Directional Deformation (Z Axis), and Equivalent Strain are illustrated below.


B: Static Structural
Total Deformation
Type: Total Deformation
Unit: mm
Time: 1
0.079687 Max
0.070833
0.061979
0.053125
0.044271
0.035417
0.026562
0.017708
0.0088541
0 Min


B: Static Structural
Directional Deformation
Type: Directional Deformation(Z Axis)
Unit: mm
Global Coordinate System
Time: 1
0.0055355 Max
-0.00086198
-0.0072594
-0.013657
-0.020054
-0.026452
-0.032849
-0.039247
-0.045644
-0.052042 Min



End of tutorial.